



Model created in COMSOL Multiphysics 6.4

Creating Modeling Domains Around a CAD Geometry

Introduction

The majority of 3D CAD files include only the geometry of the product to be manufactured. For finite element analysis, however, you find yourself often in a situation where additional geometry is needed, for example, to analyze the flow inside or outside a device. This tutorial, involving the geometry for a loudspeaker enclosure, demonstrates how to create domains inside and outside an imported CAD assembly. The following steps are covered:

- Importing a STEP file,
- Finding and deleting holes and other details from the geometry,
- Creating the union of all imported objects needed to generate a domain inside the objects,
- Using the Cap Faces operation to create a domain inside the enclosure,
- Creating a domain that surrounds the imported enclosure.

For the step-by-step instructions of this tutorial, see the book *Introduction to Design Module*.

Application Library path: Design_Module/Tutorial_Examples/
loudspeaker_modeling_domains
